CFD Modeling of Aerosol Entry in Simplified Smoke Detector Geometries

JAMES A. IERARDI and Prof. JONATHAN R. BARNETT
Center for Firesafety Studies
Worcester Polytechnic Institute
Worcester, MA 01609, USA

ABSTRACT

The interaction between a ceiling jet and smoke detector is being investigated with computational fluid dynamics (CFD) modeling in order to better under aerosol entry phenomena. The CFD modeling is performed in two stages. The first stage simulates the bulk transport of smoke from the fire to the detector location. The second stage uses the ceiling jet conditions from the first stage as an inlet boundary condition. The geometry of a smoke detector and the fluid region in its immediate vicinity are modeled in order to examine the nature of the aerosol entry for the given inlet condition and geometry.

NIST's Fire Dynamics Simulator (FDS) is used in the first stage to simulate a specified fire under a 3m (9.84ft) high unconfined ceiling. The fires of interest are 50kW at steady state and include wood, polyurethane foam, and heptane fuel sources. These fuels have been selected because they are included in smoke detector approval standards such as UL 268 and EN 54-7. The environmental conditions in the ceiling jet for detector placement 9.14m (30ft) on center are specified as output from the FDS model. The variables of interest in the ceiling jet are velocity, smoke density, and temperature and are used as an inlet boundary condition for the second stage of CFD modeling.

AEA Technology's CFX-TASCflow was used in the second stage to model simplified smoke detector geometries and the fluid region in the immediate vicinity. A baseline scenario of the fluid region with no detector specified was modeled in order to have reference values of velocity, smoke density, and temperature. Comparisons were made between results from the detector geometry and the baseline scenario to examine conditions outside and inside the detector profile.

One limitation of the current approach is the lack of aerosol dynamics sub-models to address particle agglomeration, deposition, and sedimentation. Neglecting such phenomena in the CFD modeling provides results that under-predict particle size distribution and over-predict particle number and mass concentrations.

ACKNOWLEDGEMENTS

This research has been made possible through the financial support of NIST's Building and Fire Research Laboratory (Grant # 70NANBOH0023), Hughes Associates, Inc., as well as System Sensor and Notifier Fire Systems.

KEYWORDS: Aerosol entry, computational fluid dynamics, smoke detectors, smoke entry